AS4-3501 Tutorial

This tutorial will demonstrate how to setup the material properties and run the necessary analysis modules for an AS4-3501 case.

Note: To see the **Solution** and results for this tutorial, please import the **AS4-3501_Complete** package in the **Solutions** library.

Setup

1. Select **New Project** under **Project** menu. Skip this step if you reached here by clicking on AS4-3501 Test Validation Tutorials link from MCQ Welcome page.

2. Make sure **Millimeter-Second-Newton** is selected under the **Unit System** drop down list box located near the upper right corner of the screen

3. Select **Save As** option under **Project** menu. You may name the project as: **AS4-3501**

Fiber/Matrix/Ply Calibration

Usually in-plane ply properties, fiber volume fraction, and fiber tensile modulus are available to composite analyzers. The analysis will help obtain complete set of in-situ fiber and matrix properties that will result in the desired ply properties. The analysis will also predict the unknown out-of-plane properties.

4. Right click on **Analysis** in the tree and select **Add->Fiber/Matrix/Ply Calibration.**

5. Right click on the **Calibration Input** option and select **Import from File.**

6. Browse for **AS4-3501_input_data_SI_Units.txt** file under the **Input** directory

7. Double click (or) right click and select **Edit** on **Calibration Input** option and verify the values and units, as shown in Figure below

Select Material Type

- (selected) Ply Non-Glass Fiber
- Ply Glass Fiber

Description	Units	Initial Value	COV
Manufacturing Data			
Fiber Volume Ratio		6.0000E-01	0.05
Void Volume Ratio		2.0000E-02	0.05
Manufacturer Fiber Properties			
(Ef11) Longitudinal Modulus	N/(mm^2)	2.3100E+05	0.1
Manufacturer Matrix Properties			
(Em) Young's Modulus	N/(mm^2)	4.4100E+03	0.1
Unidirectional Test Tape Ply Properties			
(E11) Longitudinal Modulus	N/(mm^2)	1.2600E+05	
(E22) Transverse Modulus	N/(mm^2)	1.1000E+04	
(G12) Shear Modulus	N/(mm^2)	6.6000E+03	
(NU12) Poisson's Ratio		2.8000E-01	
(S11T) Longitudinal Tensile Strength	N/(mm^2)	1.9500E+03	
(S11C) Longitudinal Compressive Strength	N/(mm^2)	1.4800E+03	
(S22T) Transverse Tensile Strength	N/(mm^2)	4.8000E+01	
(S22C) Transverse Compressive Strength	N/(mm^2)	2.0000E+02	
(S12S) In-Plane Shear Strength	N/(mm^2)	7.9000E+01	

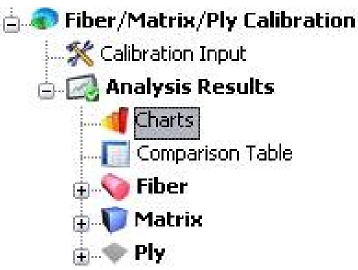
8. Make sure **Ply Non-Glass Fiber** is selected as AS4- fibers are usually assumed transversely isotropic

9. Modify these values if need to do so and note the fiber and void volume ratio values of 0.60 and 0.02, respectively, as initial approximate values

10. Since there is variation in the reported values from the vendors, 5% coefficient of variation (COV) is assigned to the fiber and void volume ratios and 10% for the fiber and matrix modulus. The user can change these COV values as see fit for their case

11. Right click on **Fiber/Matrix/Ply Calibration** option and select **Run Analysis.**

12. After a moment, the **Analysis Results** will appear in the tree.



13. Double click on **Charts** and **Ply**, **Fibe**r and **Matrix** tabs to see/verify the in-situ fiber and matrix properties obtained from input unidirectional lamina test properties earlier

14. Right click on **Fiber** in the tree and select **Export to Input Materials**

15. Name the fiber as **AS4-**

16. When prompted to export the **Fiber Volume Ratio**, click **Yes**

17. Right click on **Matrix** and select **Export to Input Materials**

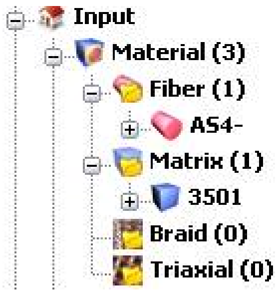
18. Name the matrix as **3501**

19. When prompted to export the **Void Volume Ratio**, click **Yes**

20. Similarly, right click on **Ply** and select **Export to Input Materials**

21. Name the ply as **AS4-3501**

22. The exported material will appear under the Material tree, as shown in the Figure below



23. Collapse the Fiber/Matrix/Ply Calibration tree to hide it for now.

Ply Mechanics

Verify ply properties from in-situ fiber and matrix constituent properties. The user can vary the fiber and void volume fraction at this time to predict and study their influence on the overall ply properties

24. Right click on **Laminate (1 Ply)** and select **Edit**

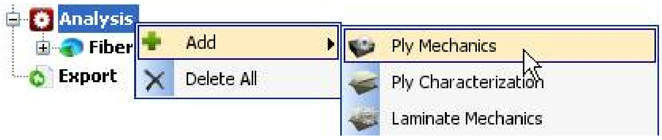
25. Make sure one ply is defined using AS4- and 3501 matrix and correct Fiber and Void volume ration from previous analysis

26. Change **Ply Thickness** to be **0.254 mm**

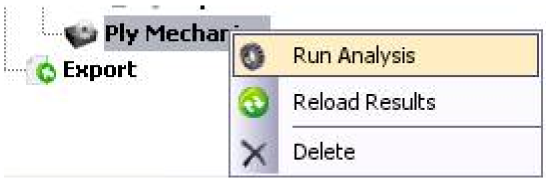
27. Select the ply row

28. Right click on it and select **Insert a Row After** to make a second row so the layup has **two** ply entries

29. Right click on **Analysis** and select **Ply Mechanics**



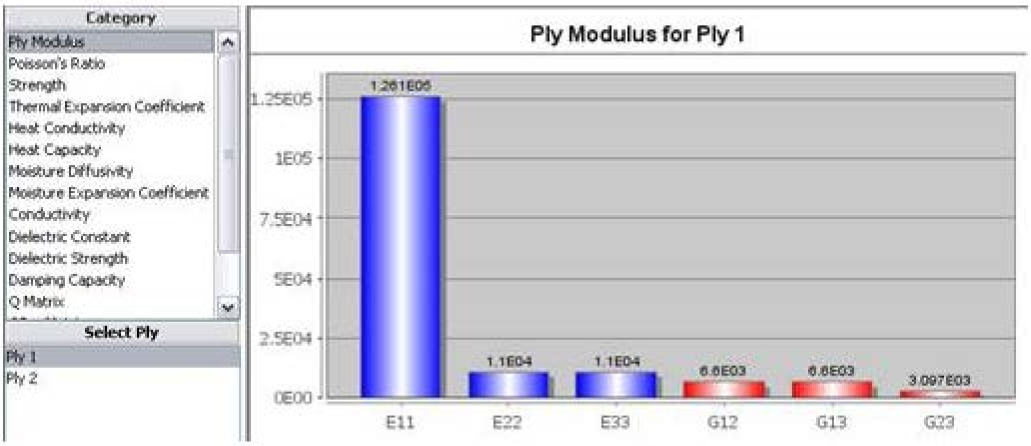
30. Right click on **Ply Mechanics** option and select **Run Analysis**



31. **Analysis Results** will appear after the analysis is finished

32. Double click on **Charts** to see/verify the ply properties obtained from in-situ fiber and effective matrix properties.

Note: These values should be similar to your original input tested ply properties. Ply Mechanics analysis will show additional values for missing directions (e.g., **E33, G23, G13, v23, v13, S13, S23, Q matrix, QBar matrix, S matrix, and SBar matrix**, etc)

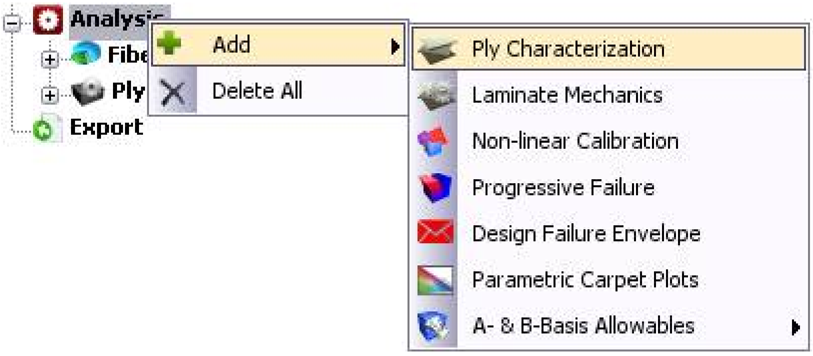


33. Be sure that Ply 1 and Ply 2 both show similar material properties as they are identical in a two ply laminate created earlier. You can select Ply 2 under Select Ply panel

Ply Characterization

Graphically verify the variation in ply properties, dominant failure zones with variation in fiber and void volume fraction and loading direction orientation with respect to the fibers

34. Modify/Add **Failure** option if need to do so. Leave default for this exercise. The analysis works only at lamina level; therefore, modification of the laminate layup option will be ignored for this option
35. Right click on **Analysis** in the tree and select **Ply Characterization**



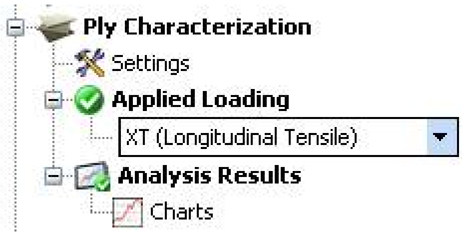
36. Double click on **Settings** option and a panel will appear. Leave the default selections for this exercise. This will result in variation in off-axis loading to vary from 0 to 90 degree with an increment of 5 degree.

    Note: you can select fiber or void volume ratio instead of Unidirectional Angle option by clicking on Unidirectional Angle.
37. Leave the default **Applied Loading** (Longitudinal Tensile).
38. Right click on **Ply Characterization** and select **Run Analysis**

39. **Analysis Results** will appear after the analysis is finished

40. Double click on **Charts** to see/verify the Ply properties obtained from in-situ fiber and matrix properties. The mode allows you to view various ply properties under Arbitrary or Principal Coordinate System as a function of ply angle, fiber, or void volume fraction.

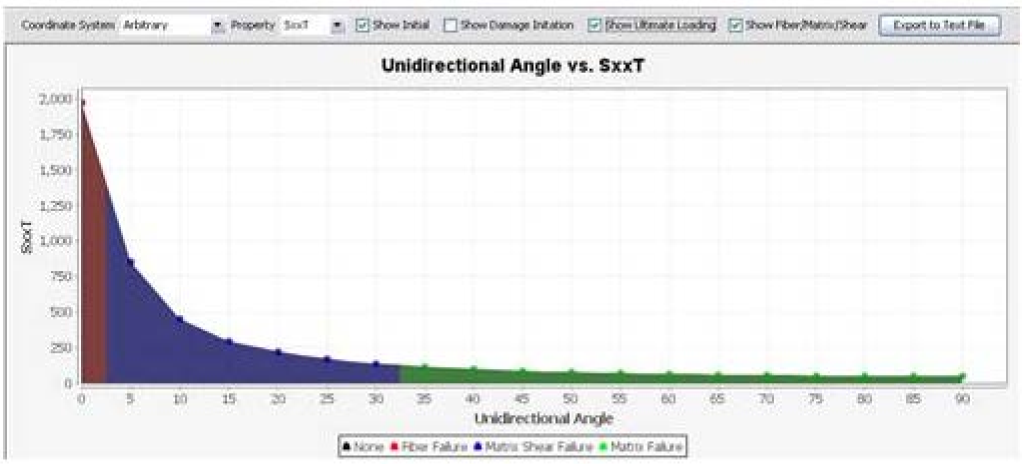
41. Select **Arbitrary** under **Coordinate System** drop down list box above graph area. Make sure **Exx** is selected under **Property** drop down list.



42. Make sure **Exx** is selected under **Property** drop down list. Figure below shows variation in the modulus with variation in 0ff-axis loading direction.

43. Select **SxxT** under **Propert**y drop down list.

44. Check **Show Ultimate Loading** and **Show Fiber/Matrix/Shear** options located near the top of the graph area. Figure below shows the dominant failure criteria switching with the off-axis loading for a ply.



Laminate Mechanics

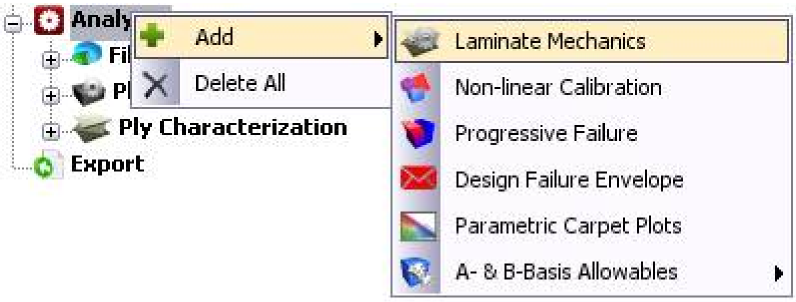
Predict laminate level material properties using in-situ fiber/matrix, ply, or matrix properties as input along with braid cards for fabric, woven or 3D architecture. The analysis relies on progressive failure analysis, micro-mechanics and classical laminate theory.

45. Double click on **Failcrit_1** option under **Failure** to verify and accept the default failure criteria selection

46. Right click on **Laminate (1 Ply)** and select **Edit**

47. Make sure **FailCrit_1** failure option is selected for each of ply entry row

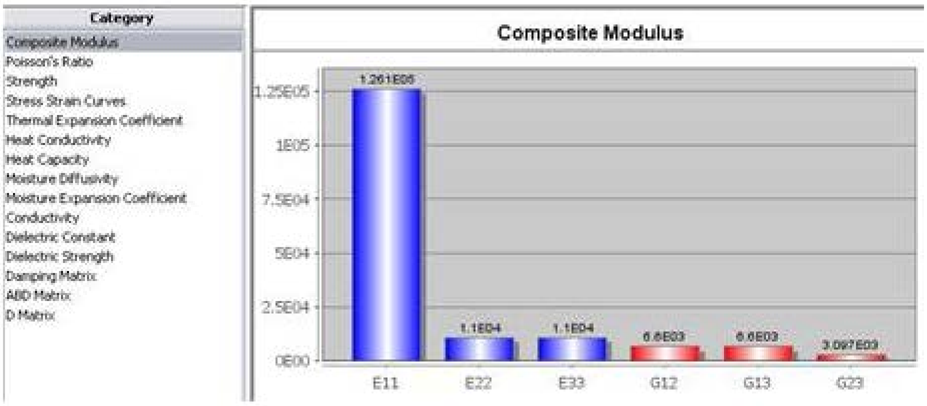
48. Right click on **Analysis** and select **Laminate Mechanics**



49. Right click on **Laminate Mechanics** option and select **Run Analysis**

50. **Analysis Results** will appear after the analysis is finished.

51. Double click on **Charts** to see/verify the Laminate properties obtained from in-situ fiber and matrix properties. These values should be similar to your original input ply properties, as we haven't changed the plies in the laminate definition. The code will show additional values for missing directions (e.g., **E33, G23, G13, v23, v13, ABD matrix**, etc)



52. In addition to **Ply Mechanics** Category panel, **Laminate Mechanics Category** panel shows the **Stress-Strain Curves** option. Selecting this option allows the user to see the stress-strain curves for the laminate in longitudinal tension and compression, transverse tension and compression, and in-plane shear loading conditions.

53. Select different **Loading Directions** from the drop down list box above **Results** panel

54. Note that in-plane shear (SXYS) loading direction stress-strain curve is linear. Most tests indicate that this result is highly nonlinear, as we will see in following steps.

Non-linear Calibration

Predict the in-situ matrix stress strain curve from in-plane shear ASTM standard test data (e.g., ASTM D5318). The analysis can be used to reverse engineer any fiber or ply nonlinearity as well. The analysis relies on progressive failure analysis, micro-mechanics and classical laminate theory.

55. Modify/Add the **Failure** option if need to do so. For this exercise, the plies are all 0 degrees (default from before). This works at lamina/laminate level; therefore, modification of the laminate option will be ignored for this option.

    Note: If you has access to the axial stress-axial strain data from ASTM D3518 that contains the [+/45]ns, then you have to update the Laminate definition with exact layup.

    Note: if you have reduced shear stress-shear strain data for a 0 degree ply (like this exercise), then you have to use 0 degree plies as the Laminate.

56. Right click on **Analysis** and select **Non-linear Calibration**

57. Double click on **Material Setup** under **Non-Linear Calibration** and a panel will appear.
58. Make sure **3501** matrix is selected next to Name Entry
59. Select **XYS** for shear **Loading**,
60. Select **SS** for shear strength of the matrix, as shear test depends on matrix shear strength.
61. Set **1.1** for **Strength Upper-Bound Factor**.

    Note: You may have to increase this for other materials if calibration quits without fully calibrating with test shear stress curve.

62. Select **EPSXY** for **Strain Type** as test data to be compared is for shear stress-shear strain.

63.Select **SIGXY** for **Laminate Stress Type** as test data to be compared is for shear stress-shear strain.

Name	Value
Material	
Type	MATRIX
Name	3501
Loading	XYS
Loading Steps	100
Strength	SS
Strength Upper-Bound Factor	1.1
Stiffness	E
Strain (X-AXIS)	
Strain Mode	LAMINATE
Strain Type	EPSXY
Stress (Y-AXIS)	
Stress Mode	LAMINATE
Laminate Stress Type	SIGXY

64.Double click on **Test Data Curve** under **Non-Linear Calibration**

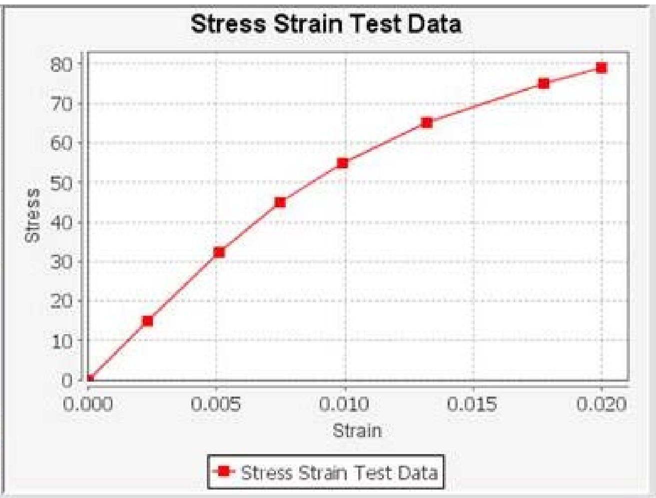
65.Right click on table in the panel to show the popup menu and select **Read from File** option to read the "**test_curve_SXY_EPSXY_0_deg_plies.txt**" test data file supplied with the exercise in the working directory.

Note: This shear stress shear-strain curve for a single ply can be derived from ASTM D3518 or Isopescu type in-plane shear test data

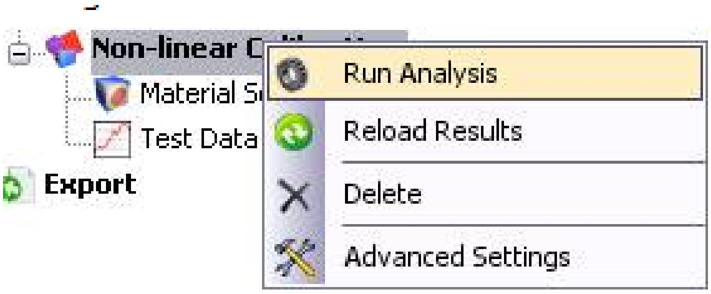
	Strain	Stress
1	0.0	0.0
2	0.002277	15.0
3	0.005072	32.5
4	0.007456	45.0
5	0.009907	55.0
6	0.013187	65.0
7	0.01772	75.0
8	0.02	79.0

Options

Downsample Number of Points



66.Right click on **Non-linear Calibration** and select **Run Analysis**



67.Once the analysis finishes the **Analysis Results** window will appear

68.Double click on Calibrated SS Curve option. This is the reverse engineered in-situ matrix curve that causes the lamina to show nonlinear behavior under shear loading, as shown in Laminate shear stress-shear strain curve from the imported test data.

69.Right click on the **3501** matrix and select **Export to Input Materials** to export it back to the Input materials with the new SS Curve in it.

Note: If you have calibrated different temperature data you can click on Add Temperature Set button and enter the new temperature data. The default is room temperature 21.11 degree Celsius (73 degree Fahrenheit).

70.Click on **Yes** when prompted to overwrite existing 3501 matrix properties. Since the material properties are same, this step will only export the calibrated in-situ matrix stress-strain curve under the Material option under the tree.

Re-Run Laminate Mechanics

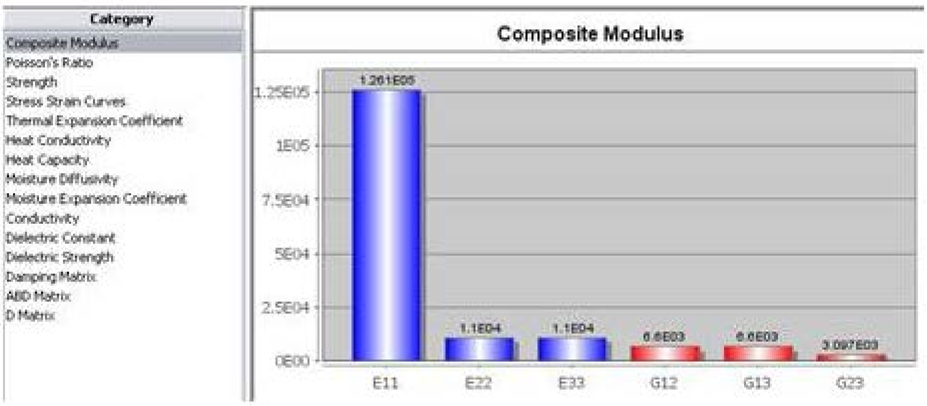
We will now verify the stress-strain curve predictions against test data for all loading directions using the calibrated in-situ nonlinear stress-strain data for 3501 epoxy in the previous section.

71. Right click on **Laminate Layup** option and select **Set Current**. This will make sure that the desired layup is being analyzed when multiple layups are defined.

72. Right click on **Laminate Mechanics** option and select **Run Analysis**

73. **Analysis Results** will appear after the analysis is finished.

74. Double click on **Charts** to see/verify the Laminate properties obtained from in-situ fiber and matrix properties. These values should be similar to your original input ply properties, as we haven't changed the plies in the laminate definition.

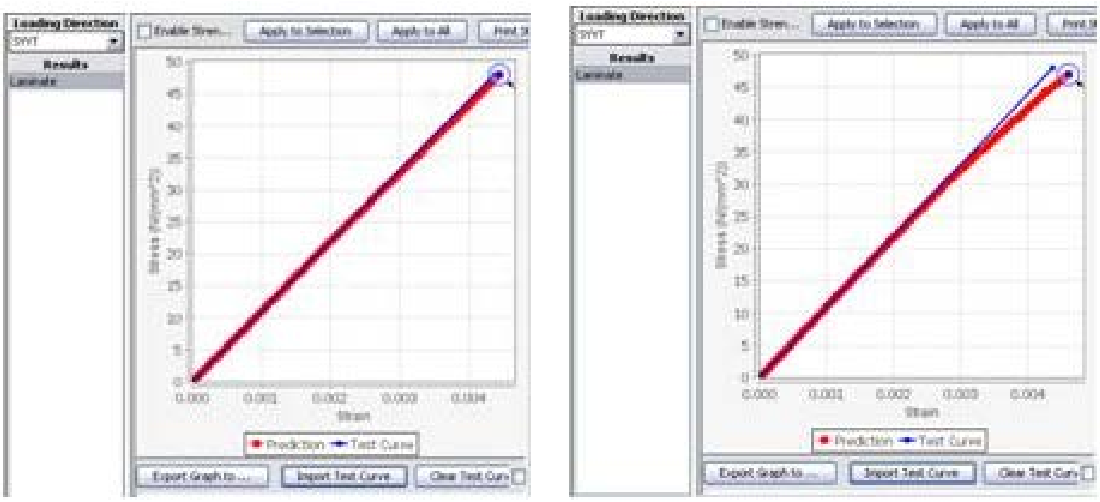


75. Select **Stress-Strain Curves** option under **Category** panel. Selecting this option allows the user to see the stress-strain curves for the laminate in longitudinal tension and compression, transverse tension and compression, and in-plane shear loading conditions.

76. Select different **Loading Directions** from the drop down list box above **Results** panel

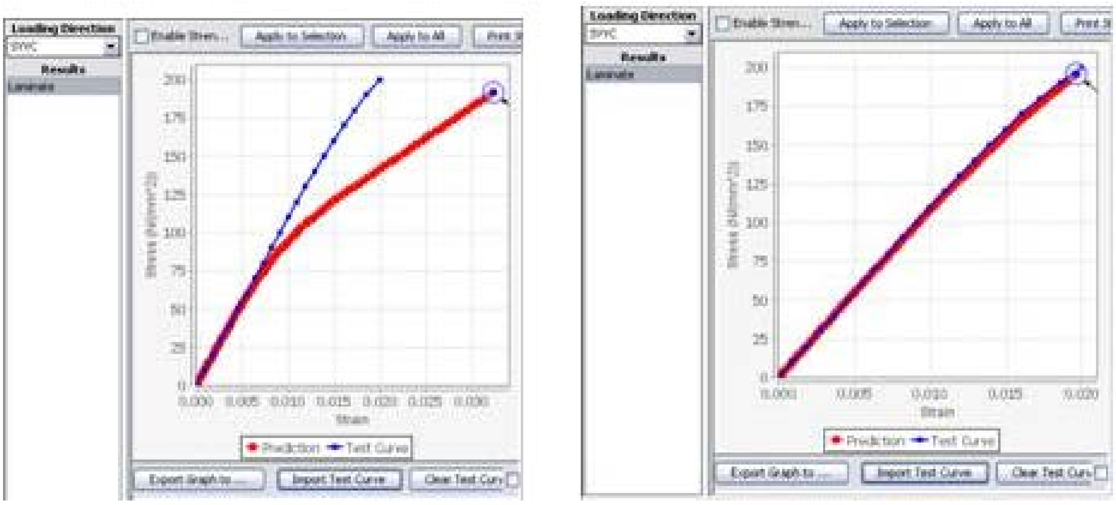
77. Note that transverse tension and compression, and in-plane shear (SXYS) loading direction stress-strain curves are nonlinear unlike before nonlinear calibration analysis.

78. You can import the test stress-strain curves in the working directory for other loading cases one-by-one to verify if simulated stress-strain curves match the test curves in all 5 loading directions.



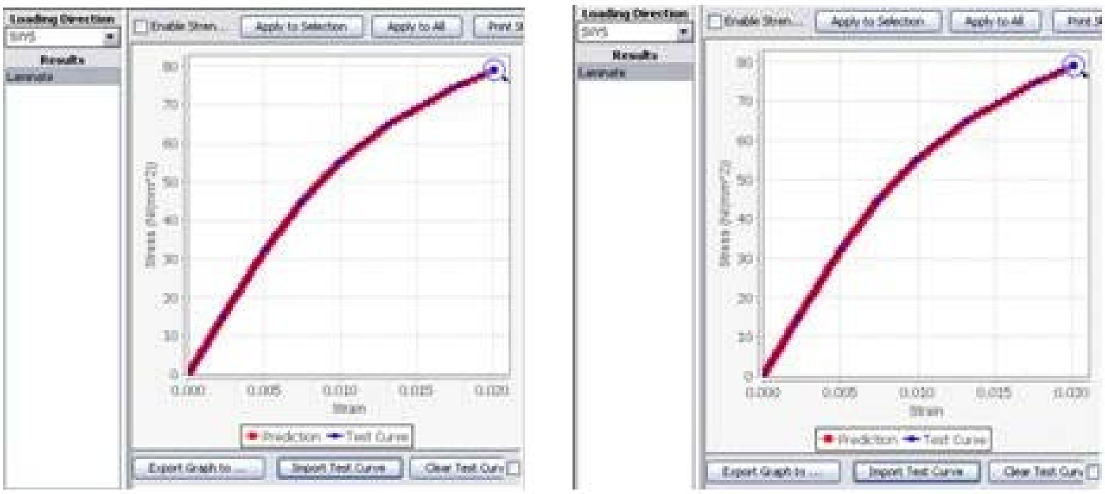
(a) Pressure Constant = 0 (b) Pressure Constant = 1.8

Transverse Tension Stress Strain curves



(a) Pressure Constant = 0 (b) Pressure Constant = 1.8

Transverse Compression Stress Strain curves



(a) Pressure Constant = 0 (b) Pressure Constant = 1.8

In-Plane Shear (shear stress vs shear strain) curve

In-plane shear (shear stress vs shear strain) curves

79. Expand **3501** matrix under **Material** tree

80. Double click on **Stress Strain Curve**

81. Enter **1.8** for **Pressure Constant** text box above the tabulated stress-strain data.

    Note: The pressure constant term is used with Modified Von Mises theory to account for difference in yield strength in polymeric materials under tension, compression and shear loading.

Note: The pressure constant is set to zero for isotropic metallic materials and for polymeric prepreg layers from 1.0 to 2.5. It is recommended that this value be calibrated by increasing in the 1.2, 1.4, 1.6,… The higher the pressure constant the lower the transverse tension yield strength and vice versa for transverse yield strength

Note: The longitudinal tension and compression, and in-plane shear stress-strain curve will be nearly unaffected from any changes in the pressure constant term.

82. Rerun the **Laminate Mechanics** analysis. The updated stress-strain curves are shown on the right side of the previous figures.

83. We will extract the strain limit values for the ply from simulated stress-strain curves from the corresponding five loading curves.

Ply Strain Limit	Value
Eps11T	1.570E-02
Eps11C	1.186E-02
Eps22T	4.600E-03
Eps22C	2.010E-02
Eps12S	2.009E-02

84. Right click on **Strain Limits** option under the tree

85. Select **Add->StrainLimit**

86. Double click on **StrainLimit** option and enter the above listed values at correct locations and as shown below.

    Note: Leave the default 0.15 values for unknown strain limits in out-of-plane directions.

Name	Symbol	Value	Units
Longitudinal Tensile Strain Limit	EPS11T	0.0157	mm/mm
Longitudinal Compressive Strain Limit	EPS11C	0.01186	mm/mm
Transverse Tensile Strain Limit	EPS22T	0.0046	mm/mm
Transverse Compressive Strain Limit	EPS22C	0.0201	mm/mm
Tensile Strain Limit in 33 direction	EPS33T	0.15	mm/mm
Compressive Strain Limit in 33 direction	EPS33C	0.15	mm/mm
In-plane Shear Strain Limit	EPS12S	0.0209	mm/mm
Out-of-plane Shear Strain Limit	EPS23S	0.15	mm/mm
Shear Strain Limit in 13 direction	EPS13S	0.15	mm/mm

87. Click on **FailCrit_1** option under **Failure (1)** in the tree

88. Set **S12S** to **false** and **Maximum Strain Based Failure Criteria** to **true** under **Damage Criteria** tab. Note that S12S is set to false to ensure that the ply fails due to shear strain value and not because of shear stress. Shear stress can be perfectly plastic curve in some cases. Choosing S12S as one of the failure criteria may result in early failure of the ply, as soon as the yield stress is reached. If you assume linear elastic material then S12S may be left to true.

89. Click on **Critical Fracture Criteria**

90. Set **CFC (Customized Failure Criteria)** to **false** and **Maximum Strain Based Failure Criteria** to **true**

91. Double click on **Laminate (2 Plies)** option in the tree

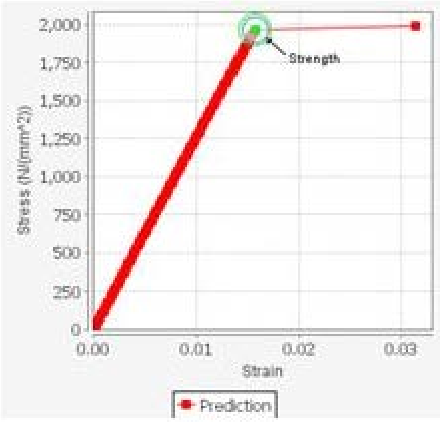
92. Select **StrainLimit** in the last column for both plies, as shown below

	Material Type	Fiber	Matrix	Temperature (C)	Thickness (mm)	Angle (Degrees)	Fiber Volume (Fraction)	Void Volume (Fraction)	Failure	Strain Limit
1	Fiber/Matrix	AS4-	3501	21.111113	0.254	0	0.570913	0.020619	FailCrit_1	StrainLimit
2	Fiber/Matrix	AS4-	3501	21.111113	0.254	0	0.570913	0.020619	FailCrit_1	StrainLimit

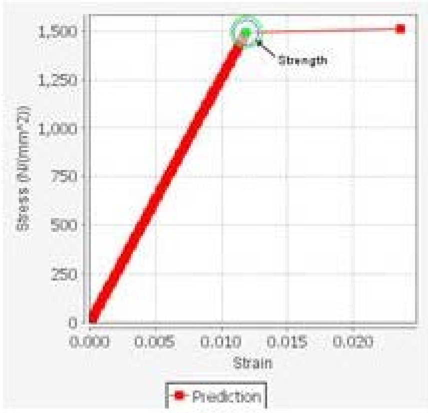
Note: You have finally calibrated the material properties and modified the failure criteria to meet your material requirements. The approach works for most cases; however, may require small adjustments depending on the material system. Please consult test validation cases for examples.

93. Rerun **Laminate Mechanics** analysis to verify the simulation results for ply are same as test data.

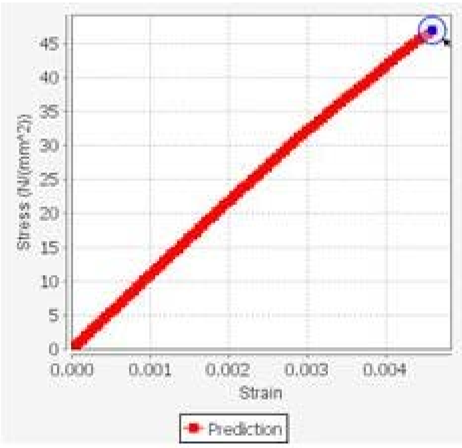
94. Please monitor the stress strain curve to judicially pick the strength values as shown below.



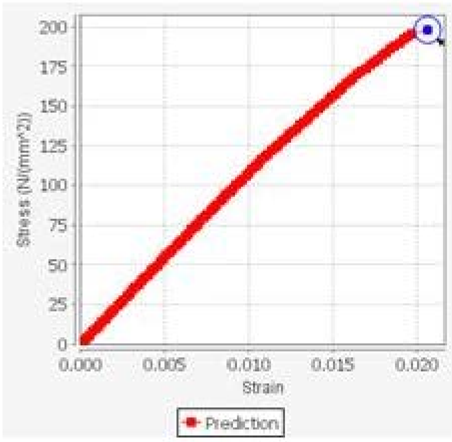
Longitudinal Tension



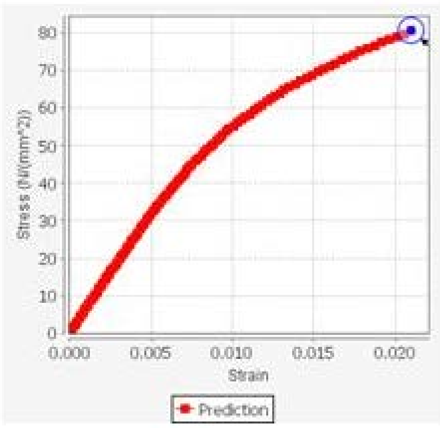
Longitudinal Compression



Transverse Tension



Transverse Compression



In-plane Shear

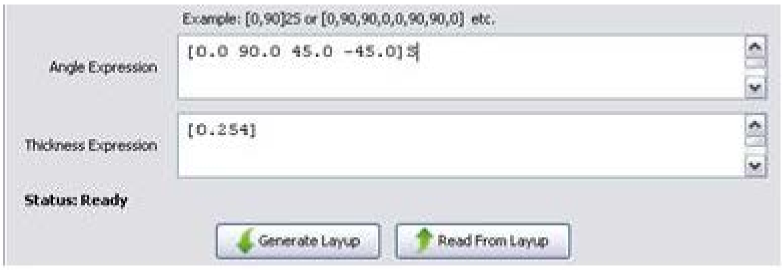
Progressive Failure

We will perform the analysis for the input using material degradation models and iterative process based on user input to ultimately predict the strength, modulus, and laminate and layer-by-layer damage evolution process.

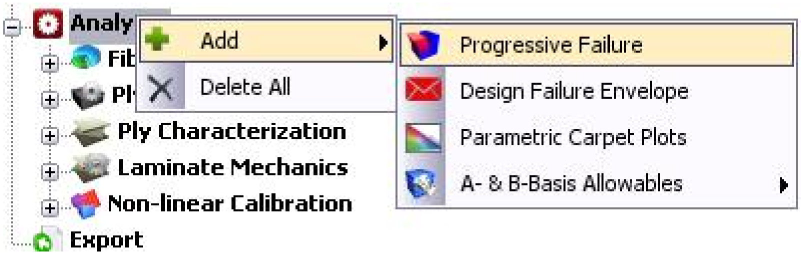
95. Right click on **Laminate (1 Ply)** and select **Expression Editor**.

96. Define symmetric quasi-isotropic layup [0/90/45/-45]S layup by writing the entry box as shown below and click on **Generate Layup** button.

    Note: You could define another layup (symmetric or non-symmetric, (or) balanced or unbalanced) using same approach



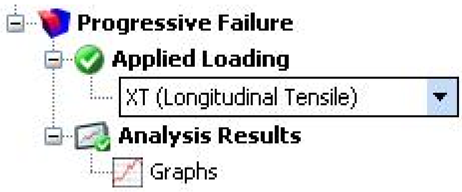
97. Right click on **Analysis** and select **Progressive Failure.**



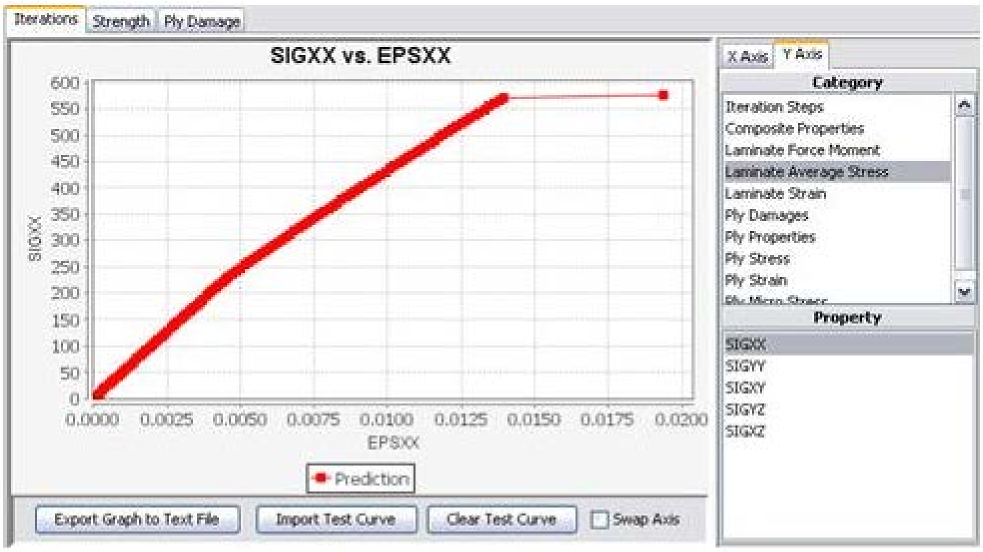
98. Make sure **Applied Loading** is set to **XT (Longitudinal Tensile)**. You can change for other loadings if need to do so.

99. Right click on **Progressive Failure** and select **Run Analysis.**

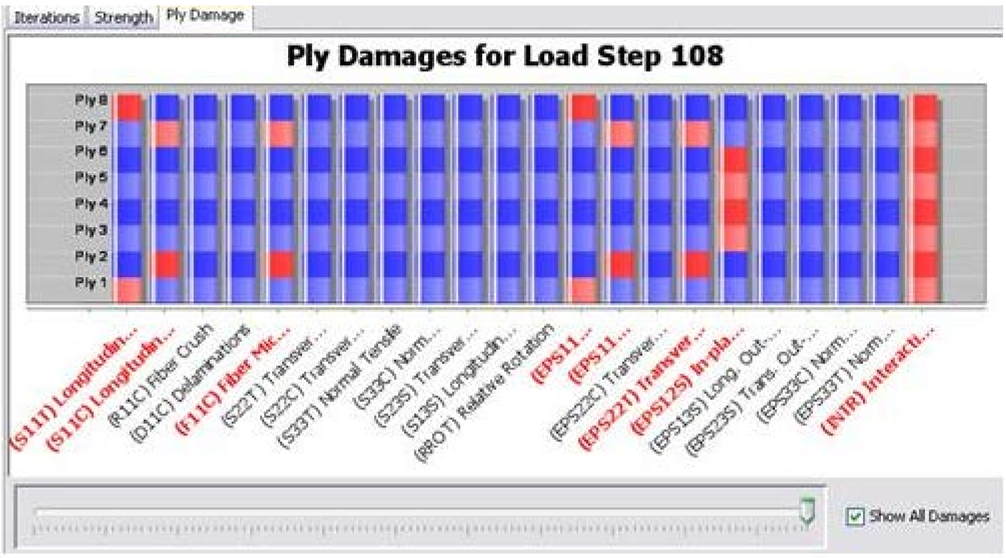
100. **Analysis Results** will appear after the analysis is finished.



101. Double click on **Graphs** to see/verify the Laminate properties obtained from effective fiber and effective matrix properties. The mode allows you to view the stress strain, damage and failure modes information at the ply and laminate levels



Laminate stress vs laminate strain

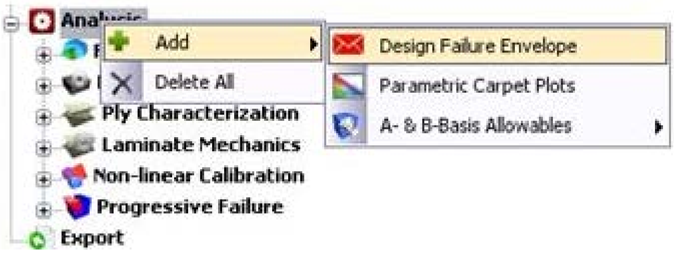


Ply-by-Ply Damage at final iteration (Ply Damage tab)

Design Failure Envelope

We will predict failure envelope for lamina or laminates based on the chosen failure criteria after the above calibration process. Works at lamina/laminate level; therefore, modification of the laminate option will be considered for this option.

102. Right click on **Analysis** and select **Design Failure Envelope.**



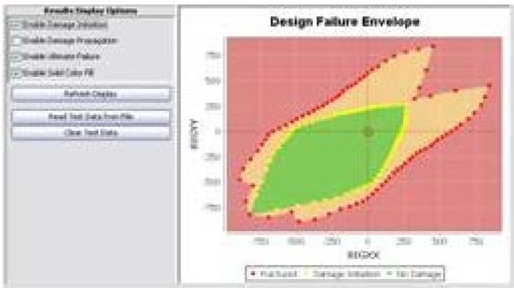
103. Double click on the **Settings** option and a panel will appear.
104. Enter **20** for **Number Quadrant Divisions** and leave the default selection for the rest of the entries. You may select different biaxial loading directions here if need to do so. Number Quadrant Divisions generate 20 data points in each quadrant, a total of 80 for the whole graph.

Settings Setup

Description	Value
X Axis	XX
Y Axis	YY
Number Quadrant Divisions	20
Division Distribution	LINEAR

105. Right click on **Design Failure Envelope** and select **Run Analysis.**
106. **Analysis Results** will appear after the analysis is finished.
107. Double click on **Charts** to see the design failure envelopes for the chosen loadings and failure criteria.

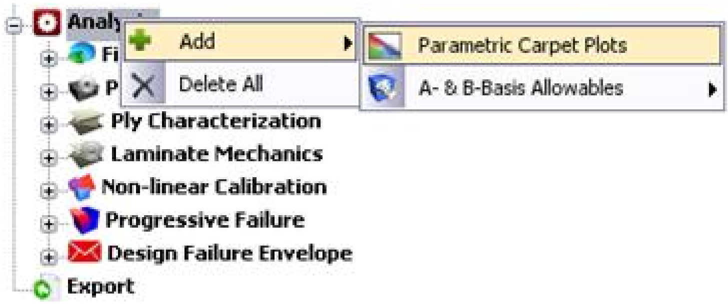
108. Click on Enable Damage Initiation and Enable Solid Color Fill options.



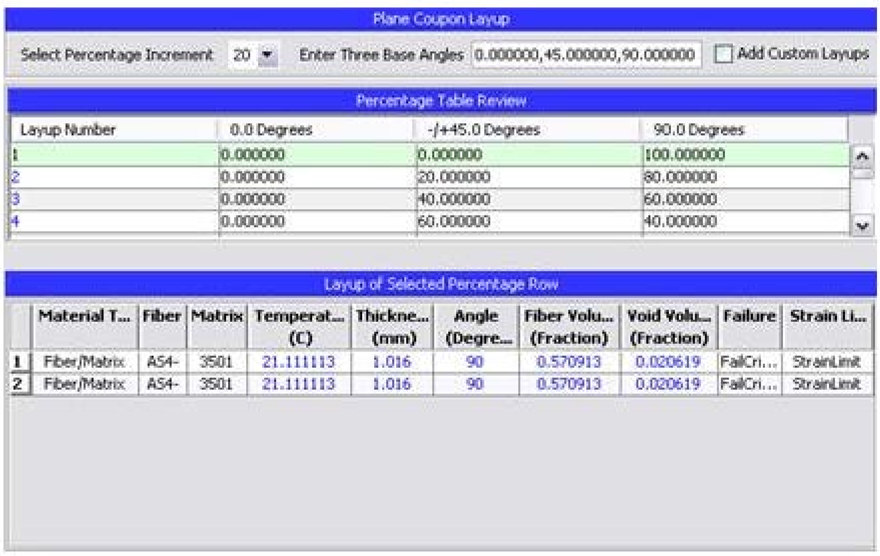
Parametric Carpet Plot

We will predict graphical representation of strength and other material properties of laminates containing symmetric and balanced plies in three different orientations

109. Right click on **Analysis** and select **Parametric Carpet Plots**



110. Double click on **Plane Coupon Layup** to show the percentage layup configuration. This will show all of the percentage layups that the analysis will run and plot in the carpet plot results. Leave the settings as is.

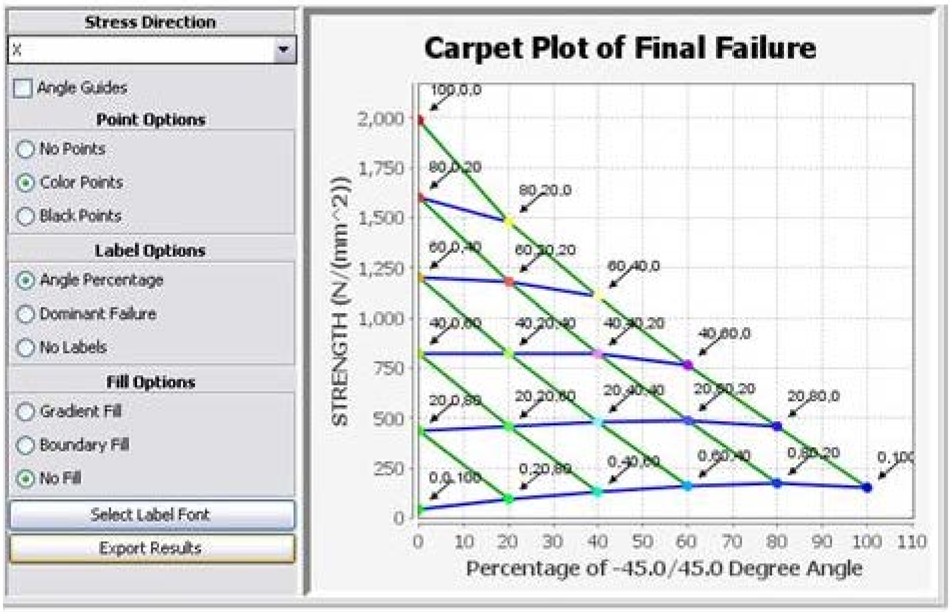


111. Leave the **Applied Loading** set to **XT (Longitudinal Tensile)**.

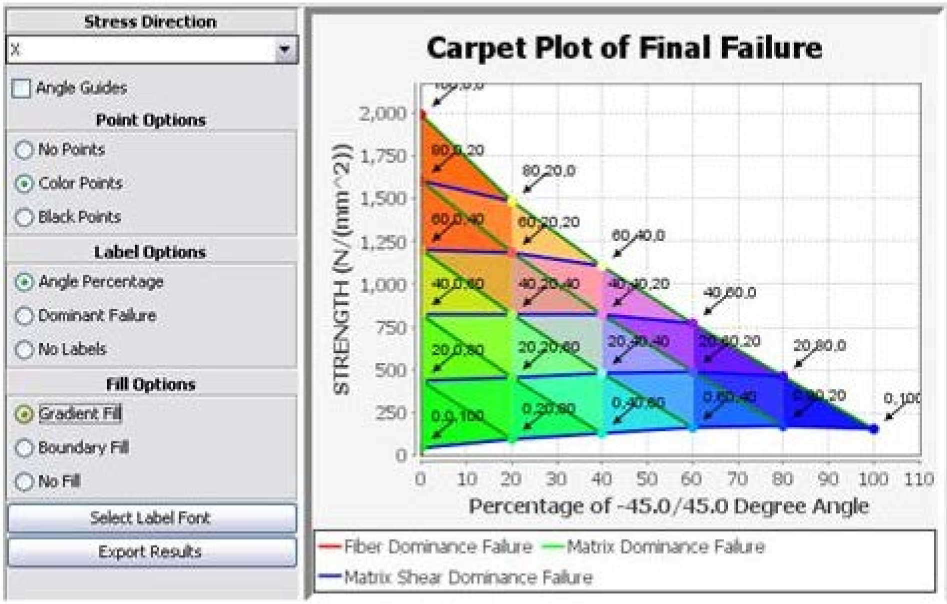
112. Right click on **Parametric Carpet Plots** and select **Run Analysis**.

113. The **Analysis Results** will appear once the analysis finishes.

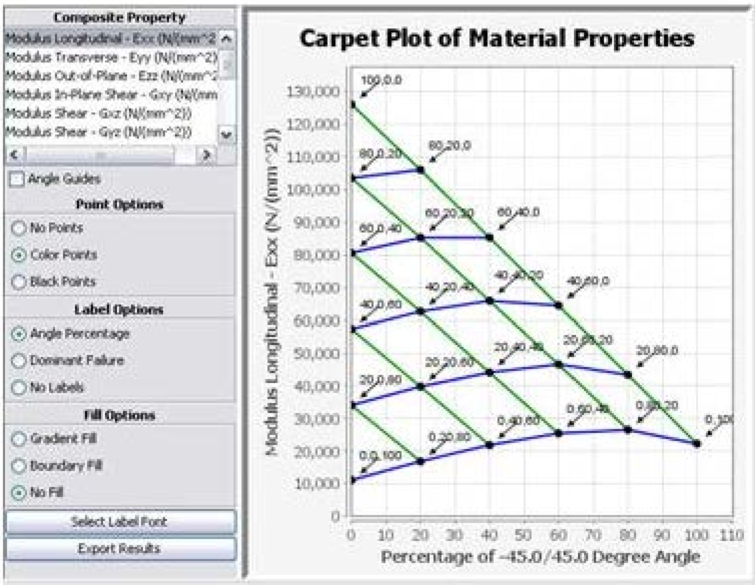
114. Double click on **Final Failure** to show the results graph below.



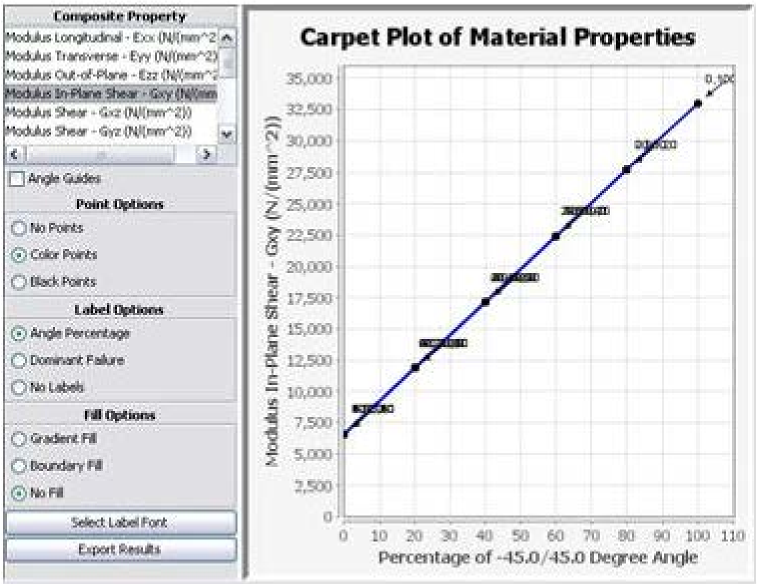
115. Select **Gradient Fill** under **Fill Options** side panel. The color marks the dominant failure criteria (fiber, matrix tension and matrix shear).



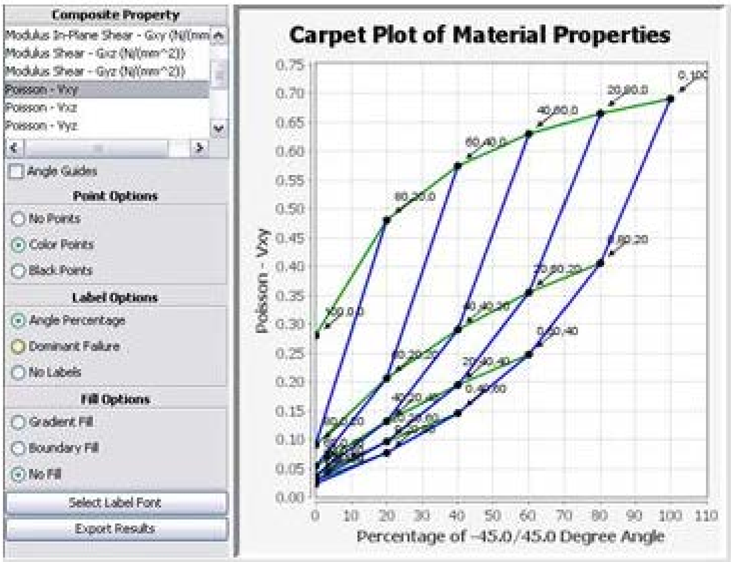
116. Double click on **Material Properties** in the tree under **Parametric Carpet Plots** analysis option to show the results graph for **Modulus Longitudinal Exx**, **Modulus In-Plane Shear Gxy**, and **Poisson's ratio Vxy**.



Longitudinal Modulus



In-Plane Shear Modulus



Poisson's Ratio Vxy

A- & B-Basis Allowables

We will predict A- and B-basis strength allowables based on material and fabrication uncertainty in the composite laminate material. The analysis will randomly vary the variables and simulate the coupon and ultimately note down the strength values from all the simulations. The strength values are then run through probabilistic analysis formulation presented in Mil-HDBK 17E and predict the A- and B-Basis Allowables. You can directly enter the scatter from the unidirectional ASTM standard tests as the variation in the constituents.

Example, Longitudinal tension and compression test correspond to the variation in Sf11T, eps11T, and Sf11C, eps11C, Transverse tension and compression test correspond to variation in SmT, eps22T and SmC and eps22C and in-plane shear test correspond to SmS and eps12S variables. SmT, SmC and SmS are in-situ matrix tension, compression and shear strength properties while Sf11T and Sf11C are in-situ fiber tension and compression strengths. While eps11T, eps11C, eps22T, eps22C, and eps12S are the longitudinal tension, longitudinal compression, transverse tension, transverse compression and shear strain limits for the ply.

The user can vary the fiber and void volume fraction based on Coefficient of Variation (COV) from manufacturing data, or assume 5% if unknown.

For this exercise we will assume all the variables to vary by 5% COV and normal distribution, except void volume fraction for which we will assume Lognormal distribution. We will also ignore test data as we do not have it available for this example. However, you can consult to other test validations for sample real input values from test case (IM7-/MT45, Tape T700/2510, and Plain Weave T700/2510)

117. Right click on **Analysis** and select **A- & B-Basis Allowables -> Simulation: (Probabilistic based predictions)**

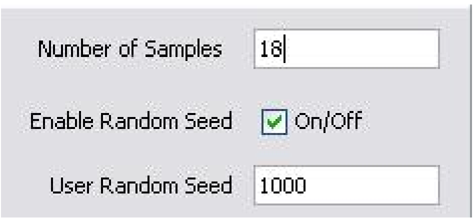
118. Double click on **Material Variables** under **A- & B-Basis Allowbles** option in the tree
119. Click on **Add** button 10 times

Material Variables Setup						
Type	Name	Property	Mean Value	Coefficient Variation	Standard Devia...	Distribution Type
FIBER	AS4-	S11T (Tensile Strength)	3.432041E+03	5.000000E-02	1.716020E+02	Normal
FIBER	AS4-	S11C (Compressive Strength)	2.415870E+03	5.000000E-02	1.207935E+02	Normal
MATRIX	3501	ST (Tensile Strength)	7.377970E+01	5.000000E-02	3.688985E+00	Normal
MATRIX	3501	SC (Compressive Strength)	3.074154E+02	5.000000E-02	1.537077E+01	Normal
MATRIX	3501	SS (Shear Strength)	1.278754E+02	5.000000E-02	6.393770E+00	Normal
STRAIN...	StrainLimit	EPS11T (Longitudinal Tensile Strain...	1.570000E-02	5.000000E-02	7.850000E-04	Normal
STRAIN...	StrainLimit	EPS11C (Longitudinal Compressive ...	1.186000E-02	5.000000E-02	5.930000E-04	Normal
STRAIN...	StrainLimit	EPS22T (Transverse Tensile Strain ...	4.600000E-03	5.000000E-02	2.300000E-04	Normal
STRAIN...	StrainLimit	EPS22C (Transverse Compressive ...	2.010000E-02	5.000000E-02	1.005000E-03	Normal
STRAIN...	StrainLimit	EPS12S (In-plane Shear Strain Limit)	2.090000E-02	5.000000E-02	1.045000E-03	Normal

120. Make sure you click **Tie Strain Limit properties Together** option. This will vary the strength and corresponding strain limits proportionally. This is important especially when small variation in shear strength can result in larger variation in the corresponding shear strain values when high nonlinearity exists.
121. Double click on **Layup Variables** in the tree under Material Variables
122. Click on **Add** button twice.
123. Select FVR (fiber volume ratio) and VVR (void volume ratio) and enter % COV and Normal and Lognormal distribution as shown below.

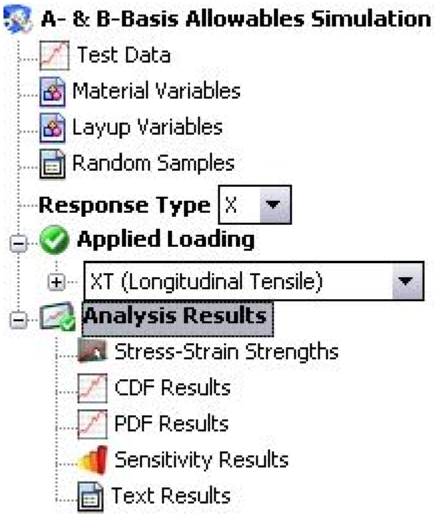
Layup Variables Setup					
Property	Plies	Mean Value	Coefficient Variation	Standard Deviation	Distribution Type
FVR	1-8	0.570913	0.087579	0.050000	Normal
VVR	1-8	0.020619	2.424948	0.050000	Log Normal

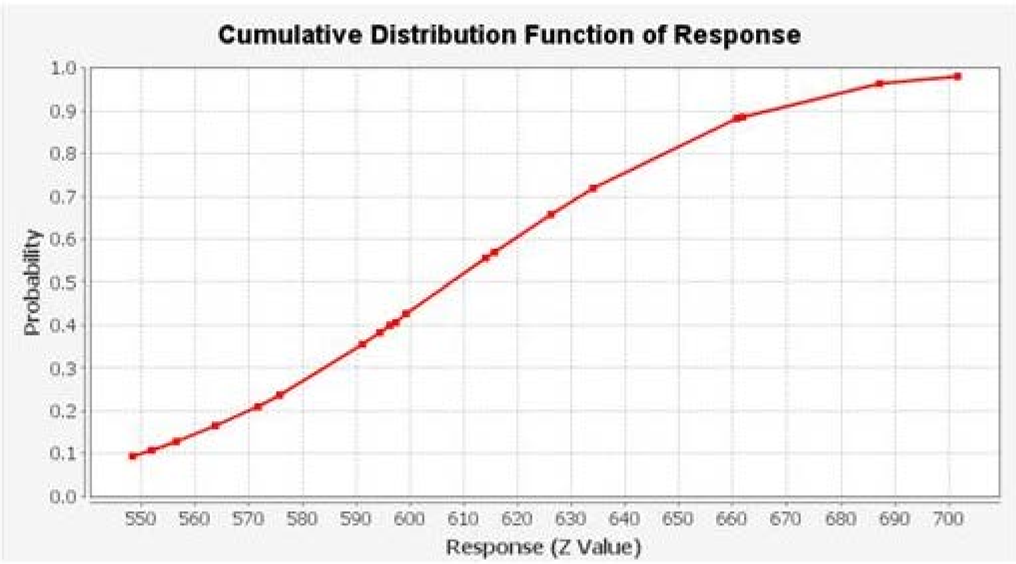
124. Double click on **Random Samples** under the tree.
125. Enter **18** for **Number of Samples** and leave the rest default

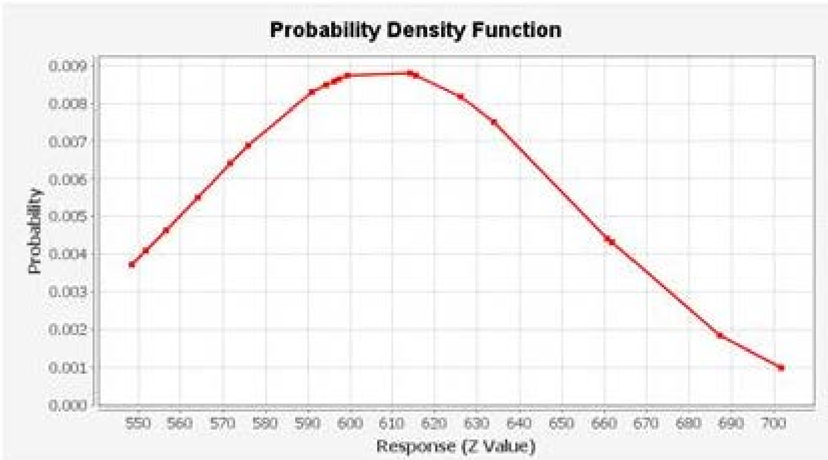


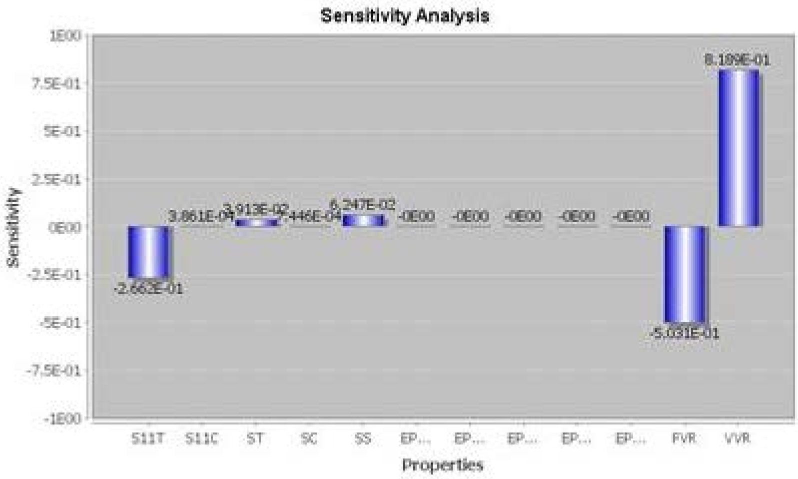
126. Right click on **A- & B-Basis Allowables Simulation** and select **Run Analysis**

127. Once the analysis is finished click on CDF Results, PDF Results, Sensitivity Results and Text Results to monitor the results.










```
========================================================
A- & B-Basis Allowables Results
========================================================
                    Mean: 607.744605
                 Std Dev: 44.898177
                     %CV: 7.387672
========================================================
Observed Significance Level (OSL): 0.268514
          Normal B-Basis: 520.245493
          Normal B/Mean: 0.856027
          Normal A-Basis: 458.112025
          Normal A/Mean: 0.753790
========================================================
Observed Significance Level (OSL): 0.382831
      Log Normal B-Basis: 526.241307
       Log Normal B/Mean: 0.865892
    Log Normal B/Norm. B: 1.011525
      Log Normal A-Basis: 475.944539
       Log Normal A/Mean: 0.783132
    Log Normal A/Norm. A: 1.038926
========================================================
Observed Significance Level (OSL): 0.057314
         Weibull B-Basis: 486.357614
          Weibull B/Mean: 0.800266
       Weibull B/Norm. B: 0.934862
```

The mean of the simulated scatters is 607.7 MPa and the predicted A- and B-Basis allowables are 458.1 MPa and 520.25, respectively. Since the OSL for normal distribution is above 0.05 we take these predicted and ignore the rest, as per the instructions from MiL-HDBK 17E.